

**ELC 4383 RF/Microwave Circuits I**  
**Laboratory 5: Circuit Tuning and Electromagnetic Simulation**

Note: This lab procedure has been adapted from a procedure written by Dr. Tom Weller at the University of South Florida for the WAMI program and from related procedures written for the WAMI Laboratory. It has been modified for use at Baylor University by Dr. Charles Baylis and teaching assistants.

This laboratory exercise is a software exercise and should be completed individually (i.e. you will not have a lab partner).

Printed Name: \_\_\_\_\_

**Please read the reminder on general policies and sign the statement below. Attach this page to your Post-Laboratory report.**

**General Policies for Completing Laboratory Assignments:**

For each laboratory assignment you will also have to complete a Post-Laboratory report. For this report, you are strongly encouraged to collaborate with classmates and discuss the results, but the descriptions and conclusions must be completed individually. You will be graded primarily on the quality of the technical content, not the quantity or style of presentation. Your reports should be neat, accurate and concise (the Summary portion must be less than one page). Laboratory reports are due the week following the laboratory experiment, unless notified otherwise, and should be turned in at the start of the laboratory period. See the syllabus and/or in-class instructions for additional instructions regarding the report format.

**This laboratory report represents my own work, completed according to the guidelines described above. I have not improperly used previous semester laboratory reports, or cheated in any other way.**

Signed: \_\_\_\_\_

**ELC 4383 – RF/Microwave Circuits I**  
**Laboratory 5: Circuit Tuning and Electromagnetic Simulation<sup>1</sup>**

**Laboratory Assignment:**

**Overview**

This software laboratory will introduce two concepts:

- 1) Using ADS to “tune” a circuit to achieve a desired performance, and
- 2) Using Momentum to perform electromagnetic (EM) simulations on a layout.

In the first part of the laboratory, you will construct a circuit schematic for a radial harmonic tuning stub. This stub should cause a high reflection to be provided to signals near 5 GHz. Harmonic tuning stubs are used to remove harmonics from a signal (i.e. at the output of a circuit) while allowing the fundamental to pass through. After constructing this schematic, you will “tune” the length of the stub until it provides the desired response (high  $S_{11}$  near 5 GHz).

In the second part of the exercise, you will generate a layout automatically from the schematic (rather than drawing the layout manually as you did in Laboratory 3). You will then simulate the layout using the Momentum EM simulator that is built into ADS.

**Procedure**

**A. Circuit Tuning Using ADS**

1. Create the circuit schematic shown in Figure 1 and save it as “**HarmonicTuningStub.dsn**”. Your substrate parameters should be the following: **H = 0.125 mm, Er = 9.6, Mur = 1, cond = 1.0E+50, Hu = 1.0E+33 mm, T = 0.005 mm, TanD = 0.0009, Rough = 0 mm**. These will also be the parameters you will use in the electromagnetic simulation part of the exercise.

Some notes:

- The width of a 50 ohm line should be determined by using the above substrate parameters in Linecalc at 5GHz.
- Set Lstub to the length of a quarter wavelength line (90 degrees electrical length) at 5 GHz on this substrate as determined by Linecalc. The stub should act as an RF short (an open circuit on the end of a quarter-wavelength line).
- Set up the S-parameter simulation to be performed from 0.5 GHz to 8 GHz in steps of 0.1 GHz.

---

<sup>1</sup> Dr. Tom Weller originally developed this laboratory procedure for the University of South Florida RF/Microwave Circuits I course and it was later modified by Dr. Charles Baylis for use at Baylor University. Parts of USF WAMI Lab Procedures 1, 2, 3, 6, and 10 were also used in constructing this assignment.

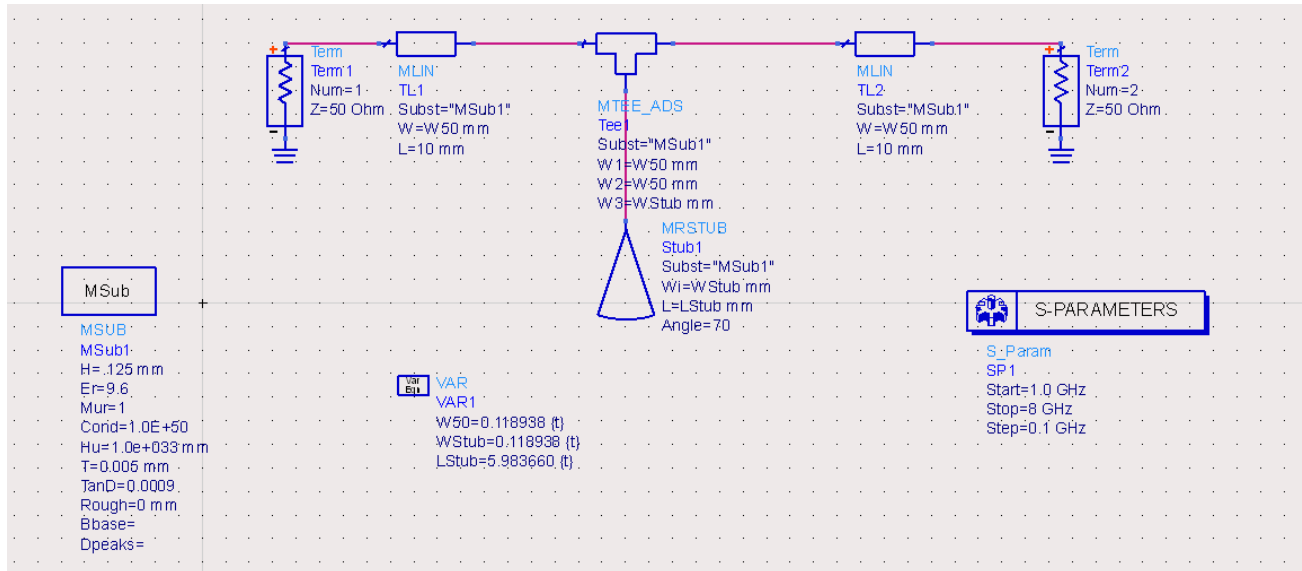


Figure 1: Harmonic Tuning Stub Circuit

2. Perform the S-parameter simulation.
3. Open a data display window and plot S11 and S21 in dB format. The results should appear similar to the following plot:

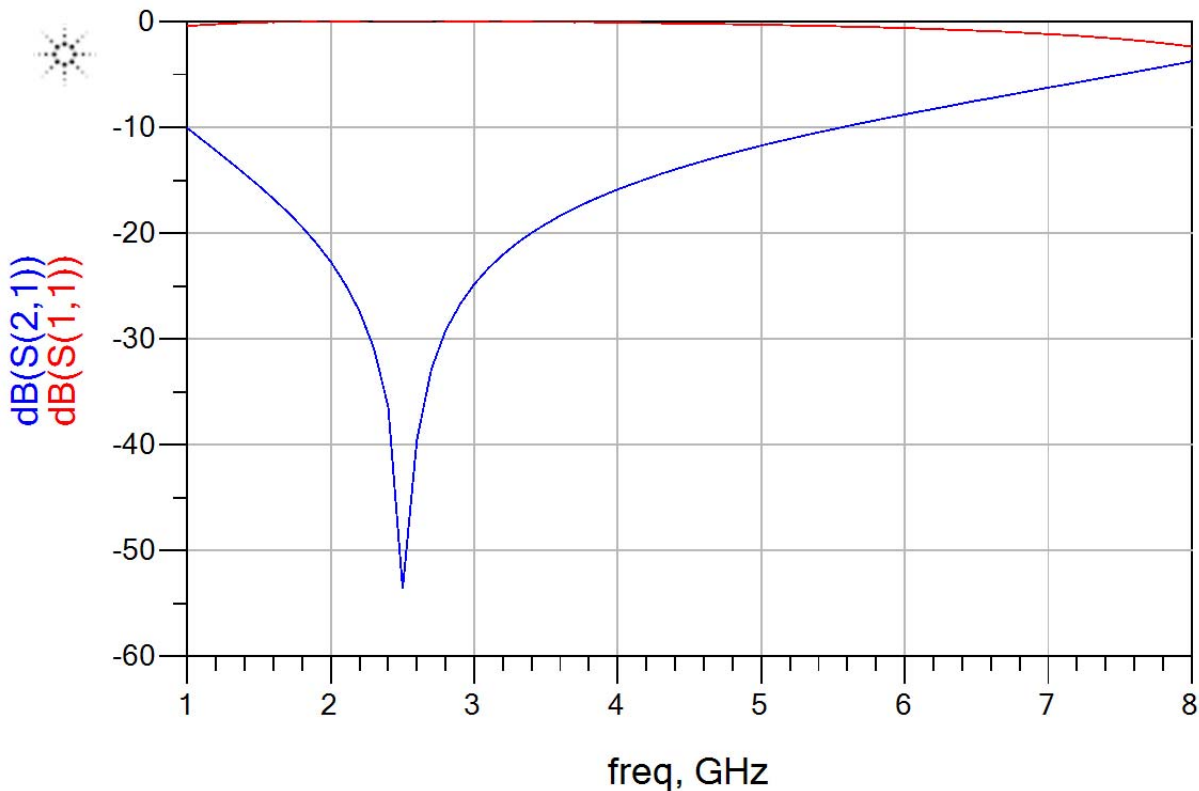


Figure 2: Un-Tuned Harmonic Tuning Stub Simulation Results

4. In the circuit schematic window, click on the Var block symbol to highlight it.

5. Go to Simulate → Tuning.
6. A tune control window will pop up as shown below. You can change the settings so that a new simulation is performed after each change, after pressing the tune button, or as the sliding bars are moved.
7. Return to the schematic and click on the var block. Click the boxes next to W50, WStub, and LStub, then click ok. This will make each of these variables available for tuning.
8. Arrange the display so that you can see the tune control pop-up window and the graph at the same time as shown in Figure 3.

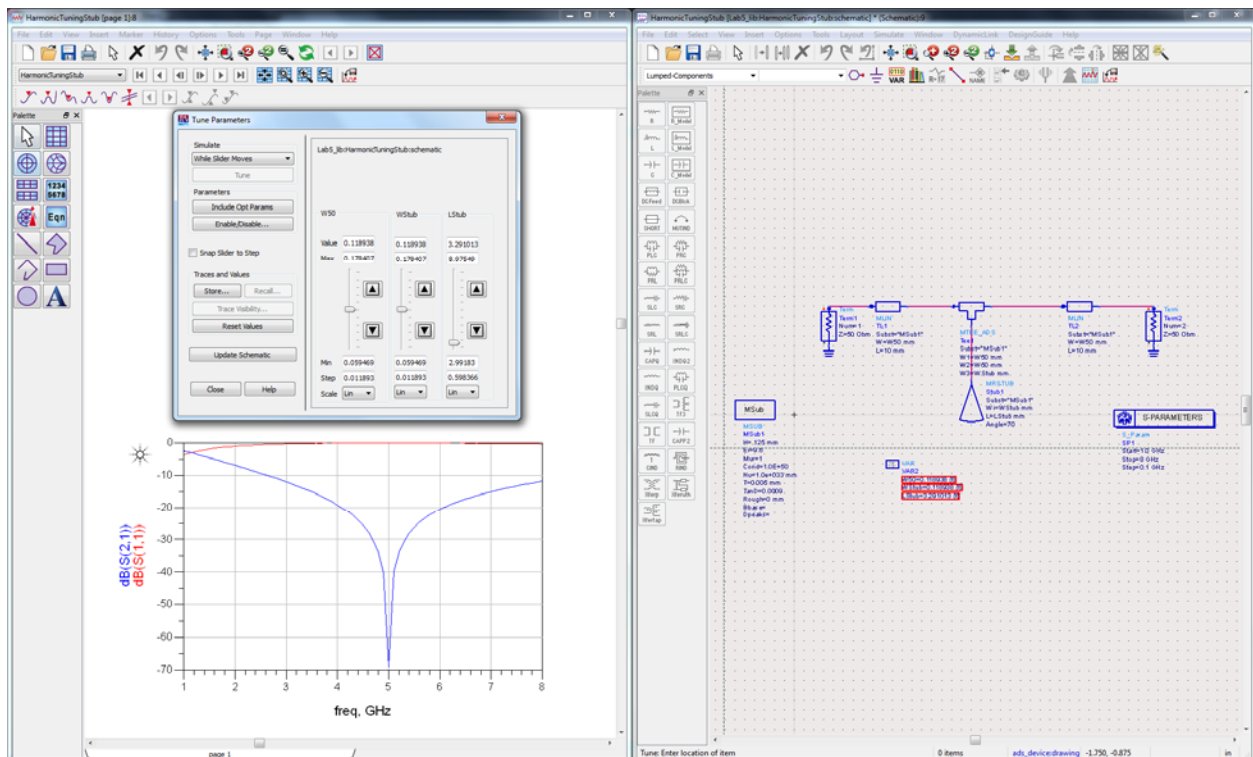
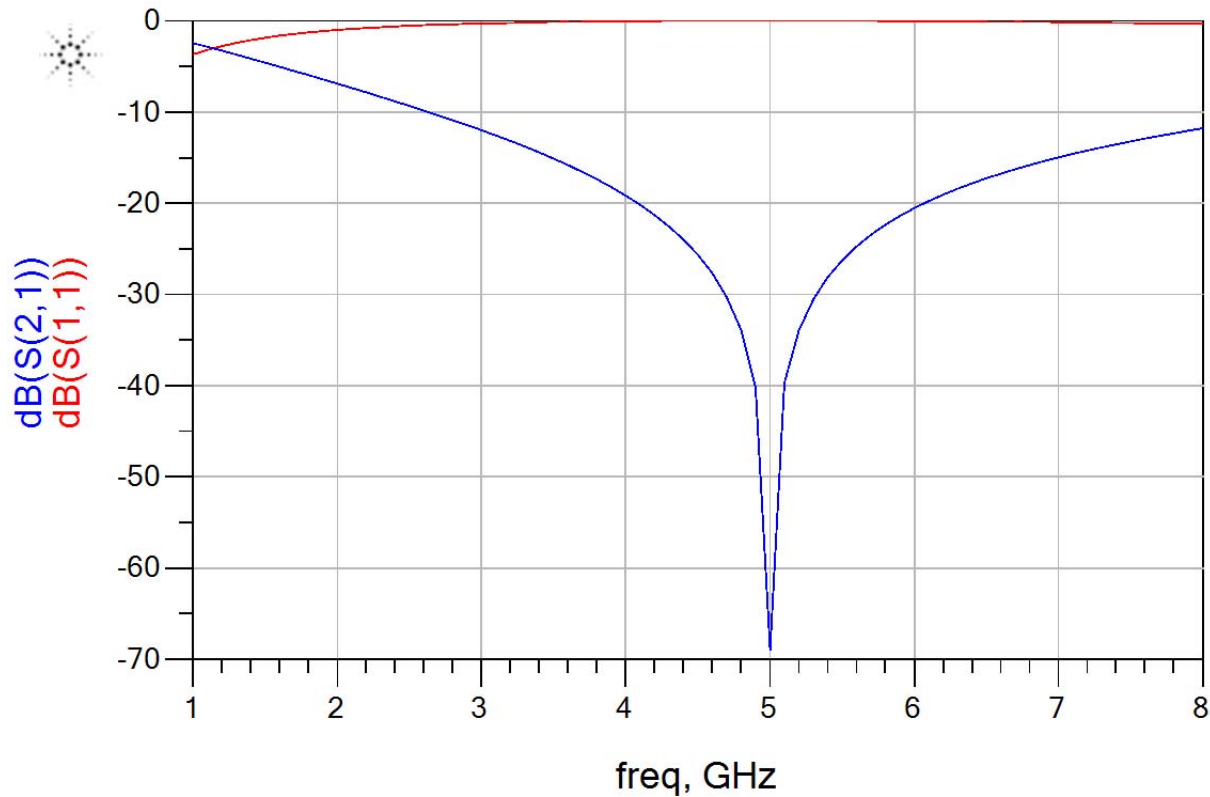


Figure 3: Circuit Tuning Windows

9. Select the option to simulate while the slider moves in the tune control window.
10. Move the slide tuner for the Lstub variable and adjust it to obtain a maximum  $|S_{11}|$  and minimum  $|S_{21}|$  at your design frequency of 5 GHz:



**Figure 4: Tuned Harmonic Tuning Stub Simulation Results**

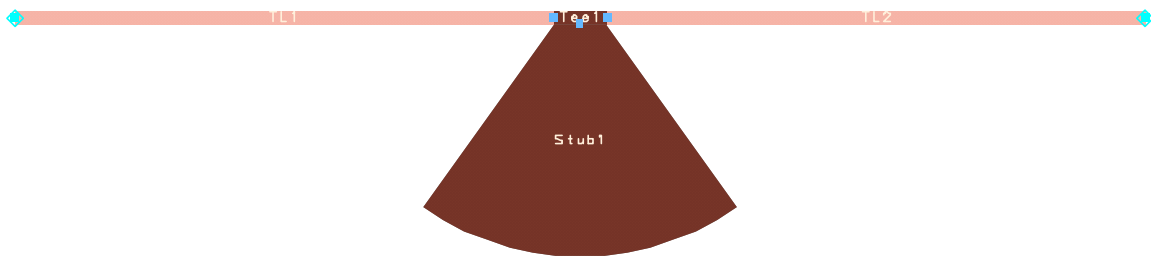
11. When you have obtained the optimum value for  $L_{\text{stub}}$ , click on the Update Schematic button in the tune control window. This will update the present values of the tunable parameters to the schematic.

12. Exit the tune control pop-up window. Generate a hardcopy of the circuit schematic and the graph showing the magnitude (in dB) and phase of the S-parameters ( $S_{11}$  and  $S_{21}$ ). Put the magnitude results into one graph and the phase results in another (you will need to create a new plot for the phase because we have not examined this yet). Save your schematic and display window. Place a copy of the schematic and both plots into your report.

## B. Layout Generation

You will now generate a layout directly from your ADS schematic using the Schematic Capture feature in ADS.

1. In the schematic window, click on Layout → Generate/Update Layout. A pop-up window will appear. It will not be necessary to modify these parameters in this case.
2. Click on OK in the pop-up window. A status report will appear describing the items that have been placed into the layout. Click on OK to close this report window.
3. The layout window should now be visible and the layout should be drawn. It should appear similar to the following:



**Figure 5: Circuit Layout**

4. Save the layout window.

### C. Momentum Simulation

1. In the layout window, click on the Port button. This button is two icons to the left of the library books icon near the top of the layout window.

2. Click on the vertex at the end of the feedline at the left side of the layout. Make sure the arrow points into the line. This will set up Port 1 for your simulations.

3. You now are ready to attach Port 2. Press Ctrl-R to rotate the arrow so that it points to the left, and attach it to the right side of the layout.

4. Save your design.

5. Select EM → Simulation Setup and select Momentum RF as your simulator.

6. Choose “Frequency Plan” on the left side. Set the sweep type to adaptive. Set the start frequency to 0.5 GHz, the stop frequency to 8 GHz, and the Sample Points Limit to 40. This is the maximum number of points that will be simulated during the adaptive simulation.

7. Select the Substrate option on the left side of the EM Simulation Window. Click New to create a new substrate and choose the Alumina substrate option. This will open a substrate window. Click on the Alumina substrate layer and change the thickness to 0.125 mm. Click the “...” next to the material dropdown box (it should say Alumina for your material) and change the real part of the permittivity to 9.6, and change the loss tangent to 0.0009. Click “Ok” to leave that window.

8. Save your substrate and close the substrate window.

9. Save your design.

10. The next step is to solve the substrate. In the “Generate” dropdown select “Substrate” and click “Go”. Momentum will perform a series of computations that are specific to the substrate definition only, not the shapes of the objects in the layout. This will take care of computations that pertain to the substrate only, so that they will not need to be repeated each time a computation is requested.

11. When the substrate calculations are complete, close the pop-up window.

12. Save your design file.

13. Now we will solve the mesh for the EM simulation. On the EM Setup Window select “Options” and then choose the Mesh tab. In most cases, the mesh frequency should be the highest frequency at which you are planning to simulate, so select the “Highest simulation frequency” option under Mesh frequency.

14. Under Mesh Density enter the Number of Cells per Wavelength (CpW) as 25. Normally, you need 20 to 30 cells per wavelength to obtain accurate results. If the number selected is too small, Momentum will not be able to accurately calculate the current distribution on the circuit, leading to poor S-parameter prediction. If too large of a number is selected, numerical errors are possible. Note that if you were to specify 1 GHz as the mesh frequency and keep the CpW the same, you would have an effective number of CpW at 8 GHz that is 8 times smaller because the physical size remains the same, but we would have specified a wavelength that is a factor of 8 times larger. The reverse is also true; if we were to specify 30 CpW at 8 GHz we would end up with 240 CpW at 1 GHz. For this example, specify 25 CpW at 8 GHz.

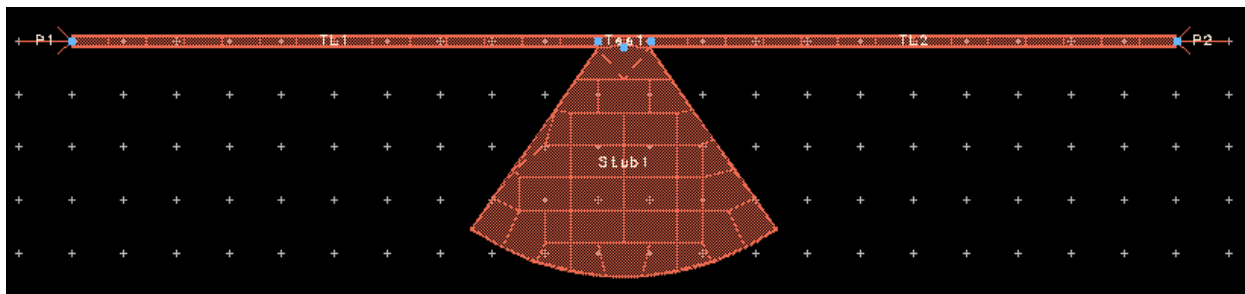
15. Select the Edge Mesh option and specify the Edge Width as 0 mm. By using Edge Mesh, Momentum will create special cells along the edges of the conductors in order to calculate the current distribution more accurately. As the frequency increases, more current is concentrated on the edges of the lines, so having more cells along the line edges improves the results of the simulation.

16. Check the Thin Layer Overlap extraction box. If desired you can read more about this option in the online documentation.

17. Select the Preprocessor tab. Under “Simplify the layout” make sure the Arc Resolution is set to 45 degrees.

18. In the “Generate:” dropdown box select “Mesh” and click “Go”. When the mesh computation is finished, close the pop-up window and save your design.

19. The layout should appear similar to the following. The mesh used for the method of moments calculations should be visible.

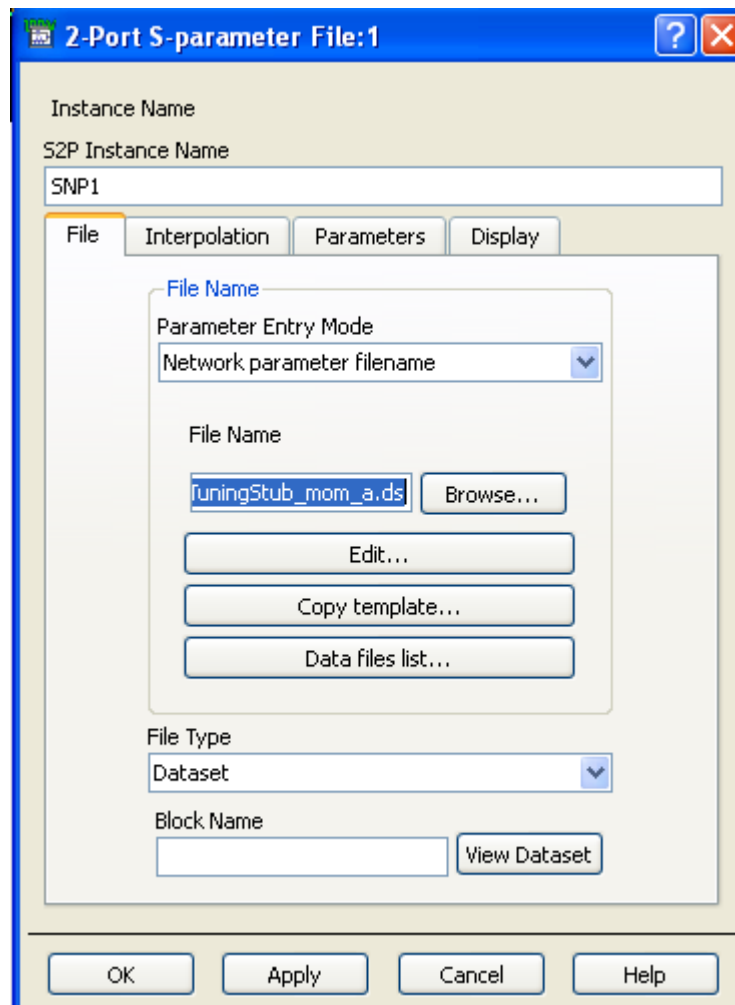


Momentum first divides the geometry into smaller components (the mesh). Following this, boundary conditions are used to calculate an approximating function for the current density in each mesh components. This is done by solving for the coefficients of approximating basis functions through a technique known as the Method of Moments.

20. In the “Generate:” dropdown box select “S-Parameters” and click “Simulate”. Momentum will simulate the circuit at the start and stop frequencies, an intermediate frequency point, or two, and then perform an adaptive simulation. The number of simulations to be performed in a given frequency region is based on how closely the results match the prediction based on the already-simulated frequency points.

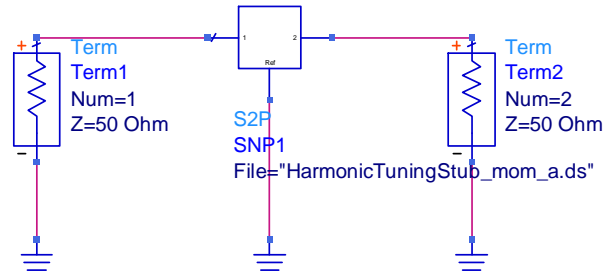
21. When the simulation is complete, a data display window will open. Create magnitude and phase plots for S21 and S11 (like what you did for the previous simulation). Place these plots in your report.

22. The name of the dataset that is created will be the same as the layout filename with an “\_a” extension, i.e. “HarmonicTuningStub\_mom\_a.ds”. This file will be saved automatically to the Data subdirectory of your project. It can be referenced from a .s2p element in an ADS schematic by selecting the file type as Dataset if desired. Create a new schematic and place a .s2p block in the schematic. Select “HarmonicTuningStub\_mom\_a.ds” as the file to be referenced. Change the file type in the selection window to from “Touchstone” to “Dataset”. Save the schematic as “MomentumData.dsn”. The dialog window for the .s2p block should appear as follows:



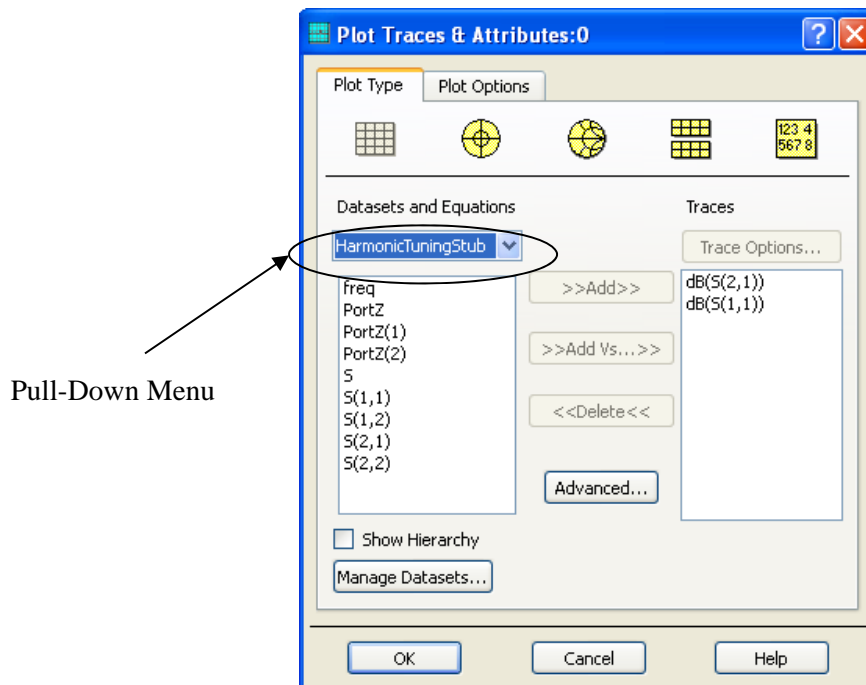
23. Place terminations on the .s2p file, insert an S-parameter block, and simulate S-parameters from 0.5 to 8 GHz in steps of 0.1 GHz. Your schematic should look similar to the following:





S\_Param  
 SP1  
 Start=0.5 GHz  
 Stop=8 GHz  
 Step=0.1 GHz

24. Generate a graph in the data display window that contains both the circuit simulation and Momentum simulation results. Generate a hardcopy of the comparison graph to place in your report. You can place data from two different simulations by selecting the appropriate dataset from the pull-down menu that is shown in the following figure:



Generate a copy of the comparison between circuit and Momentum simulations for your report.  
**Report**

Your post-laboratory report should include the following:

- 1) Summary

- 2) Discussion of Results:
  - a. What is the percent discrepancy in the frequency at which the minimum S21 occurs between the circuit simulation and the momentum simulation (using the circuit simulation as the reference for the calculation)?
  - b. In general terms, how would you modify the layout/geometry so that the Momentum simulations showed the S21 minimum occurring at 5 GHz?
  - c. For the circuit simulation results, what is the percent bandwidth over which S21 is less than or equal to -20 dB?
- 3) Printout of the microstrip circuit schematic and the layout
- 4) Graph of the circuit simulation results
- 5) Graph of the Momentum simulation results
- 6) Graph comparing the circuit and Momentum simulation results.